Design Capture/DesignView to DxDesigner – FAQ Guide

This document is for information and instruction purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in written agreements between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

RESTRICTED RIGHTS LEGEND 03/97

U.S. Government Restricted Rights. The SOFTWARE and documentation have been developed entirely at private expense and are commercial computer software provided with restricted rights. Use, duplication or disclosure by the U.S. Government or a U.S. Government subcontractor is subject to the restrictions set forth in the license agreement provided with the software pursuant to DFARS 227.7202-
3(a) or as set forth in subparagraph (c)(1) and (2) of the Commercial Computer Software - Restricted Rights clause at FAR 52.227-19, as applicable.

Contractor/manufacturer is:
Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com
SupportNet: supportnet.mentor.com/

TRADEMARKS: The trademarks, logos and service marks ("Marks") used herein are the property of Mentor Graphics Corporation or other third parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the respective third-party owner. The use herein of a third- party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A current list of Mentor Graphics' trademarks may be viewed at: www.mentor.com/trademarks.

© 2012 Mentor Graphics Corporation
All rights reserved.

This document contains information that is proprietary to Mentor Graphics Corporation. The original recipient of this document may duplicate this document in whole or in part for internal business purposes only, provided that this entire notice appears in all copies. In duplicating any part of this document, the recipient agrees to make every reasonable effort to prevent the unauthorized use and distribution of the proprietary information.

Table of Contents

Overview **6**

1.1 How do Design Capture and DxDesigner Differ in History and Future Plans? *6*

1.2 Is Design Capture and/or DesignView Being Discontinued? *7*

1.3 On Which Platforms is DxDesigner Supported? *7*

1.4 How Will DxDesigner be Enhanced in the Near Future? *7*

1.5 With Design Capture I need a full schematic license for cross probing and placement. How does this work in DxDesigner? *7*

1.6 Where can I get more information about upgrading from DC and DV to DxDesigner? *8*

1.7 How can I connect with Mentor Graphics and other customers when I have a question? *8*

Constraint Management **9**

2.1 In DC and DV, rules were entered through Net Class & Net Properties. How does rule management work in DxDesigner? *9*

OTHER CES HIGHLIGHTS 9

Library & Parts Management **11**

3.1 Can the DC/Expedition Central Library be used in DxDesigner? *11*

LIBRARY TRANSLATION KEY POINTS 11

TRANSLATION PROCESS DOCUMENTS 12

3.2 Does DxDesigner support placement by symbol and by part number? *13*

3.3 Can a Parts Manager database be used in the DxDesigner flow? *14*

Schematic/Connectivity Creation **15**

4.1 What is different when wiring a schematic in DxDesigner? *15*

4.2 Are connectivity options other than traditional schematics available in DxDesigner? *15*

4.3 We do not work in a concurrent mode on projects today. What would we gain by implementing design concurrency? *16*

CONCURRENCY IN THE FLOW 16

Design Reuse and Variants **18**

5.1 Is Design Reuse supported? *18*

5.2 Does DxDesigner have an equivalent to Variant Manager? *19*

Simulation **20**

6.1 Does DxDesigner have an interface to analog analysis tools? *20*

6.2 The Design Capture interface to RF analysis tools is achieved by file transfer. Is this still possible with DxDesigner? *21*

Automation & Scripting **22**

7.1 What scripting capabilities exist for DxDesigner? *22*

7.2 What support is provided for DxDesigner scripting? *22*

Output Generation **23**

8.1 With Design Capture we use a 3rd party product to print to PDF. How is this achieved in DxDesigner? *23*

Design Outsourcing **24**

9.1 I need to send a design project to a service provider. How will DxDesigner facilitate this? *24*

Migration Process & Training **25**

10.1 Will we need different hardware or network performance to use DxDesigner? *25*

10.2 Will training be required when moving from Design Capture to DxDesigner? *25*

TRAINING OPTIONS 25

10.3 Can DC/Expedition projects be moved to DxDesigner/Expedition? *26*

TRANSLATION PROCESS DOCUMENTS 26

10.4 My company has various schematic capture products. Can these schematics be imported into DxDesigner? *27*

10.5 Today DC is used to interface with a 3rd party layout tool via the DC Connections option. Can this type of schematic be moved to DxDesigner? *27*

Index **28**

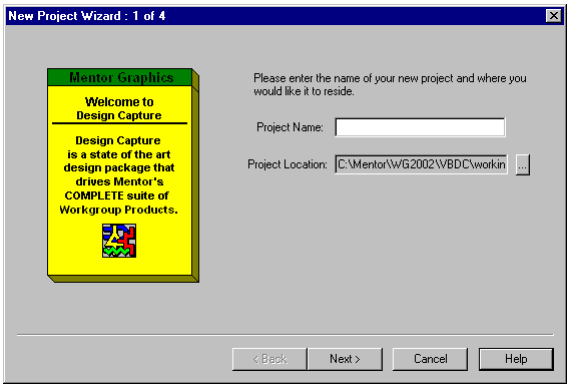
Media Contents

SHOW ME MORE: DxDataBook - Component Information Management 13

Section 1

Overview

1.1 How do Design Capture and DxDesigner Differ in History and Future Plans?


New Project Wizard : 1 of 4
Mentor Graphics
Welcome to Design Capture
Design Capture is a state of the art design package that drives Mentor's COMPLETE suite of Workgroup Products.
Please enter the name of your new project and where you would like it to reside.
Project Name:
Project Location:
C:\Mentor\WG2002\VBDC\workin
< Back
Next >
Cancel
Help

Design Capture is the legacy schematic tool developed for use with Expedition. Even though Design Capture has been successfully adopted as a schematic tool by many customers, it has architectural limitations that prevent it from supporting some modern usage models Mentor customers demand today. For this reason Design Capture will not be enhanced significantly going forward.

DxDesigner is used by thousands of users with many different layout tools. It allows scalability from an individual schematic capture tool to the concurrent enterprise design entry solution for electronic engineers. Due to its advantages it was chosen as the "go forward" design entry system for Expedition Enterprise. Since that time DxDesigner has been enhanced to improve integration with Expedition while incorporating many features from Design Capture in order to facilitate smooth migration.

DxDesigner has been significantly re-architected over multiple phases since 2008. These architectural changes enable DxDesigner to support the most modern and demanding usage models, including system-wide multi-user concurrent design. Additionally, the user interface has been refined to be more intuitive and productive, with fewer steps required to go back and forth between schematic capture, constraint entry, simulation, and layout.

1.2 Is Design Capture and/or DesignView Being Discontinued?

No. Design Capture and DesignView will continue to be supported although they will not be significantly enhanced as mentioned above. We have implemented a cross-license scheme that allows existing DC/DV users to begin working with DxDesigner while still having DC/DV available, with both tools being enabled from the same license.

1.3 On Which Platforms is DxDesigner Supported?

Windows 7 is the most popular platform however other operating systems are also supported. Details about the currently supported platforms can be found on SupportNet in the Release Highlights document for each release of Expedition Enterprise.

1.4 How Will DxDesigner be Enhanced in the Near Future?

DxDesigner is Mentor Graphics flagship product for design definition. Every release contains new technology additions, customer-requested enhancements, and continuous improvements in usability, collaboration, and flow integration.

One such addition is a new technology called DxSystems Designer, which Is being developed to define and manage multi-board systems including architectural planning, what-if scenarios, trade-offs and managing interconnect of boards with backplanes or system internal cables. DxSystems Designer is planned for the next major release of DxDesigner.

Additionally, we will continue to enhance DxDesigner with functionality simplifying the transition from Design Capture and DesignView.

1.5 With Design Capture I need a full schematic license for cross probing and placement. How does this work in DxDesigner?

Low-cost licenses have been introduced specifically for cross-probing and cross-placement in both layout and schematic: eDxDesigner and eExpedition. These tools provide easy to use dockable windows containing views of the schematic and layout, respectively. The cross-probing tools utilize design snapshots that are automatically created during forward and back annotation, protecting the “real” design against accidental or unauthorized change.

1.6 Where can I get more information about upgrading from DC and DV to DxDesigner?

With the release of Expedition Enterprise 7.9.4 the upgrade from DC/DV to DxDesigner has become easier than ever before. Our objective is to enable most of our Design Capture and DesignView customers to transition without going through a lengthy migration process. This FAQ and the resources provided through a dedicated landing page will provide all information you need to get started. This page is open to all customers, whether currently on support or not, and can be found at the link below.

http://www.mentor.com/go/DC2DX

Larger and more complex environments may require more planning and technical detail. For this purpose we provide comprehensive in-depth documentation via the migration planning resources on SupportNet. An excellent starting point is the Migration Planning Guide, located at the link below. This guide is available only to customers on support.

http://supportnet.mentor.com/reference/other-info/Expedition-Enterprise-Migration.cfm.

1.7 How can I connect with Mentor Graphics and other customers when I have a question?

Mentor Graphics customer support is available to assist you with any concerns you may have during the migration process.

In addition you can stay in touch with the migration experts within Mentor and with other customers through the Migration Community. It can be found here.

http://www.mentor.com/go/DC2DX_Connect

Use this community also for issues focused on the migration of Report Writer use cases.

Section 2

Constraint Management

2.1 In DC and DV, rules were entered through Net Class & Net Properties. How does rule management work in DxDesigner?

With the increasing speeds and complexity of topologies the traditional net classes and net properties are no longer effective to fully capture electrical and timing requirements. Today's designs require physical and electrical constraints to control signal and power integrity as timing, topology and impedance requirements. New criteria, such as mechanical and DFM requirements are managed through specific constraint types as well.

DxDesigner and the entire Expedition Enterprise flow use a spreadsheet-like Constraint Editing System known as CES, which is a significant improvement over Net Class /Net Properties. CES is also available in the DC/Expedition flow and we recommend customers to begin using CES constraints right away, gaining the benefits while in DC and preparing users for a transition to DxDesigner. A tutorial showing how to use CES to replace Net Properties/Net Classes is available on SupportNet at the link below.

http://supportnet.mentor.com/reference/tutorials/index.cfm?id=MG576733

OTHER CES HIGHLIGHTS

- DxDesigner benefits from the iCES extension to CES. iCES is a simplified interface, allowing users to enter and manage constraints on currently selected objects and to view the "actual" values from Expedition while in the schematic editor. An iCES Introduction is available at the link below: http://supportnet.mentor.com/reference/tutorials/index.cfm?id=10728

- In DxDesigner users are not required to save, compile, or package the schematic before adding constraints.

- The “DxDesigner for the Enterprise” license enables multiple engineers to concurrently add/update constraints, with CES displaying those spread sheet cells being edited by other concurrent users.
- Any constraint changes made in CES are immediately displayed to the Expedition layout designer via the Project Integration radio buttons.

Section 3

Library & Parts Management

3.1 Can the DC/Expedition Central Library be used in DxDesigner?

The DC/Expedition Central Library structure must be translated to be used by DxDesigner in the Expedition Enterprise flow. A no-cost library translator is provided in the Expedition Enterprise 7.9.* install media.

LIBRARY TRANSLATION KEY POINTS

- The translator configuration file allows users to modify translator behavior to do things like change fonts, adjust text sizes, and renamed or remove properties. The translator user guide explains the configuration file options.

- Design Capture supports compound symbol definitions with up to 8 rotational and mirrored views possible for a single symbol. Compound symbols are not yet available in DxDesigner. DxDesigner provides the ability to define alternate symbol views by assigning a numeric suffix, for example “res.1” and “res.2” may represent a horizontal and a vertical resistor. The translator provides the option

to convert DC compound symbols to uniquely numbered DxDesigner symbols, or to translate only the first view of compound symbols.

- '+', '–', '|', and '.' characters are legal in Design Capture but are not allowed in DxDesigner symbol names. The translator will replace the illegal characters with the underscore '_' character.
- DxDesigner text is slightly smaller than that in Design Capture. To achieve the same effective size the translator automatically scales up the text size (x1.25). The amount of scaling is defined in the translator configuration file.
- The DxDesigner PDB entry requires the symbol partition name as part of the symbol reference for each part. For example, a PDB entry may refer to the symbol "passives:res" where "passives" is the symbol partition in which "res" is located. It is best to remove duplicate symbols from the DC library before translation.
- The Advanced Library Editor script will help to identify and remove duplicate symbols to ensure correct PDB mapping. This script is found on the Migration Community at the following link: http://communities.mentor.com/mgcx/docs/DOC-2383
- DC/Expedition formal reuse blocks are not translated with the library and must be recreated in the DxDesigner Central Library after translating the reuse block design into DxDesigner/Expedition format.

TRANSLATION PROCESS DOCUMENTS

The translation process is fully documented on SupportNet. Key documents are listed below.

- Migration Planning Guide http://iweb.mentor.com/migration/7.9.4/index.html
- Design Capture-Expedition to DxDesigner-Expedition Translation Checklist http://supportnet.mentor.com/docs/201206008/docs/pdfdocs/dc2dx_trans_checklist.pdf

- Design Capture/Expedition PCB to DxDesigner/Expedition PCB Translation Guide
  http://supportnet.mentor.com/docs/201206008/docs/pdfdocs/dc2dx_trans_process.pdf
- Design Capture to DxDesigner translation process tutorial
  http://supportnet.mentor.com/reference/tutorials/index.cfm?id=10574
- Design Capture to DxDesigner Concepts and Terms Cross Reference
  http://supportnet.mentor.com/reference/appnotes/index.cfm?id=10854

3.2 Does DxDesigner support placement by symbol and by part number?

Yes, DxDesigner continues to support these instantiation methods from the Central Library, adding an enhanced selection filter and graphical symbol/cell preview. Formal Reuse Blocks are also instantiated from this same dialog.

DxDesigner also supports property-based part searching from a corporate component information system. This is a provided through DxDataBook, an integrated part selector that works with any ODBC-compliant data source. A quick introduction of DxDataBook can be found at the link below.

SHOW ME MORE:
DxDataBook - Component Information Management

A basic DxDataBook setup tutorial can be found at the link below.
http://supportnet.mentor.com/reference/tutorials/index.cfm?id=10600.

Customers with more sophisticated library needs may prefer to use DMS, the enterprise solution for library and data management. Information is available at the link below.

http://www.mentor.com/products/pcb-system-design/library-data-management/dms/.

3.3 Can a Parts Manager database be used in the DxDesigner flow?

Parts Manager Administrator and client applications are replaced by DxDataBook in the DxDesigner flow. The Parts Manager database cannot be used directly by DxDatabook. Data may be extracted from Parts Manager in order to create a new database for use with DxDatabook. A process guide describing how to convert Parts Manager data to DxDatabook can be found at the link below.

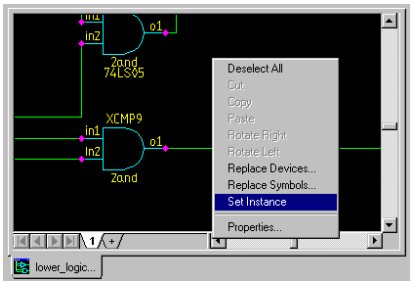
http://communities.mentor.com/mgcx/docs/DOC-3200

Section

4

Schematic/Connectivity Creation

4.1 What is different when wiring a schematic in DxDesigner?



DxDesigner and Design Capture are very similar but there are some differences as described below.

- No “save” is required in the DxDesigner environment. The database is constantly saved, with backup and rollback available.

- The wiring connectivity model is different between the two applications. For example, a named net can connect to another same-named wire at the same level of DxDesigner hierarchy without an off sheet connector. If desired, DxDesigner DRC rules can be set up to enforce DC-like rules, for example to issue a warning if a named connector is not present.

- DxDesigner requires a separate global tap for each power/ground signal defined. When translating DC schematics it may be necessary to replace a single global tap with individual global taps for each signal.

- Other terminology differences are described in the following application note on SupportNet: http://supportnet.mentor.com/assets/MG576018/10854.pdf.

4.2 Are connectivity options other than traditional schematics available in DxDesigner?

DxDesigner has a spreadsheet-like connectivity option known as “Interconnect Table” (ICT). An ICT describes the contents of a hierarchical block utilizing components

chosen from the Central Library and connectivity added in the spreadsheet cells. A read-only documentation graphic can be automatically created if required.

A tutorial can be found at the following link.
http://supportnet.mentor.com/reference/tutorials/index.cfm?id=10594.

4.3 We do not work in a concurrent mode on projects today. What would we gain by implementing design concurrency?

Design concurrency is a change in methodology that allows design teams to work together on the same design at the same time rather than taking turns contributing in serial fashion. Changes are communicated dynamically between engineering, design teams and the analysis tool users.

The chief benefit across the flow is a reduction in design cycle time as multiple engineers and layout designers can work in parallel without waiting on one another to perform transitional steps such as compiling a database.

Communication is improved since engineers work on the same database and can see one another's work. Design reviews can happen at any time since all the data is always available.

Data integrity is improved and validation efforts are reduced since all users work on a single database without having to break the design and apart and stitch it back together.

CONCURRENCY IN THE FLOW

Concurrency is enabled across the Expedition Enterprise flow as described below.

DxDesigner

- Multiple engineers can work on the same schematic at the same time, even within the same sheet.

CES

- Multiple specialists can enter design constraints for their domain of expertise at the same time, and can see one another's work instantly.

- Constraints can be entered early in the design process without having to package the schematic or choose a physical stackup.
- Actual results from the physical layout can be viewed and compared to the desired constraint values at any point in time.

Expedition XtremePCB

- Layout designers can work simultaneously on one design database and can see one another's work instantly.
- Project Integration flags indicate when changes are available from the schematic or CES. Designers can continue to work concurrently while changes from DxDesigner or CES are forward annotated.

Section 5

Design Reuse and Variants

5.1 Is Design Reuse supported?

Yes, DxDesigner supports both “informal” and “formal” reuse methods. Both methods have been enhanced far beyond what is possible with Design Capture.

Key features of DxDesigner reuse:

- Informal reuse is supported by the ability to copy sheet from one project to another with options to retain the copied referenced designators or to reset the reference designators in the new design.
- Formal reuse capability in DxDesigner supports “logical only” reuse blocks as well as “logical & physical” reuse blocks. Both block types are managed in the Central Library and maintain constraints along with the design data.
- The application of formal reuse blocks into designs has been improved to provide the following capabilities:
    - Constraints are merged from reuse blocks into the host design.
    - Tools easily identify out-of-date reuse blocks and allow users to update them from the Central Library.
    - When instantiating a reuse block in DxDesigner it is possible to remap layers, bus contents and global nets to match those in the host design.
    - Reference designators within reuse blocks can be automatically renumbered with a prefix, suffix, or “as needed”.
    - Reuse blocks can be “dissolved” into the host design if desired, where they become part of the design with no relationship back to the original block in the Central Library.

Additional information about design reuse in the Expedition Enterprise flow can be found on SupportNet at the link below.

http://supportnet.mentor.com/reference/technotes/public/technote.cfm?id=MG564725.

5.2 Does DxDesigner have an equivalent to Variant Manager?

Yes, DxDesigner provides a Variant Manager that surpasses that provided with Design

Capture. Key features include:

- Variant Manager support Physical variants and Function Managed Variants.
- Variant Manager allows users to choose alternate parts from the DxDatabook or DMS selection tools.
- Reuse blocks can be replaced in variants like any standard component.
- DxDesigner displays unique variant views without having to create multiple schematics.
- "Do not place" parts can be removed from variant views in DxDesigner, Expedition, FabLinkXE, and Drawing Editor.
- Unique assembly BOMs are automatically created for each variant.

For more information in Variant Manager refer to the User's Manual at the link below.

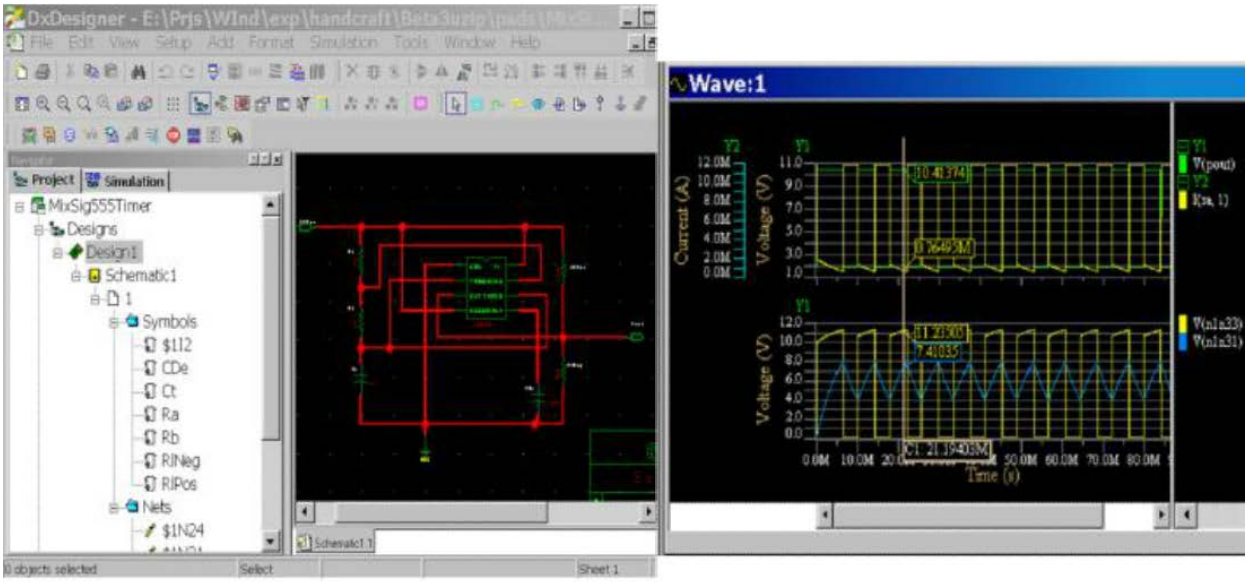
http://supportnet.mentor.com/docs/201206008/docs/pdfdocs/eevm_user.pdf.

Section

6

Simulation

6.1 Does DxDesigner have an interface to analog analysis tools?



Yes. DxDesigner has tight integration with HyperLynx Analog, which has its own integrated command toolbar within DxDesigner. Similar to Design Capture and Design View, it allows users to create one common schematic for PCB design and simulation.

HyperLynx Analog has its own built-in simulation engine and can be extended to use the Eldo and AMS analysis engines.

6.2 The Design Capture interface to RF analysis tools is achieved by file transfer. Is this still possible with DxDesigner?

DxDesigner provides tight integration with Agilent ADS and AWR Microwave Office, facilitating bi-directional design transfer and co-simulation. The user can design in DxDesigner and then launch the analysis engine to verify the RF group. DxDesigner can also directly import RF structures from the RF tools.

Section 7

Automation & Scripting

7.1 What scripting capabilities exist for DxDesigner?

DxDesigner includes built-in Automation capabilities enabling customers to tailor functionality to their own needs. Scripts can be written in a variety of languages. Technical information on the DxDesigner Automation capabilities can be found at the link below.

http://supportnet.mentor.com/reference/technotes/public/technote.cfm?id=MG71758.

7.2 What support is provided for DxDesigner scripting?

DxDesigner is supported like all other aspects of DxDesigner – questions and issues can be addressed to customer support. Support will not write scripts for you, but they will answer questions, provide guidance, and log issues.

The Automation and Scripting Community serves as a communication hub, with Mentor and customers sharing scripts and advice with each other.

http://communities.mentor.com/mgcx/community/pcb/automation_scripting.

Section 8

Output Generation

8.1 With Design Capture we use a 3rd party product to print to PDF. How is this achieved in DxDesigner?

DxDesigner includes DxPDF, a generator that creates searchable PDF files, displaying symbol and net property values as a tool tip for extended collaboration. When block hierarchy is defined, DxPDF enables the reader to descend into the block similar to how you would in DxDesigner.

You can find additional details about the PDF generation from DxDesigner on SupportNet at the link below.
http://supportnet.mentor.com/reference/technotes/public/technote.cfm?id=MG572785.

.

Section 9

Design Outsourcing

9.1 I need to send a design project to a service provider. How will DxDesigner facilitate this?

- The Expedition Undock/Dock capability enables users to send the Expedition layout to a remote site and to later synchronize it back into the DxDesigner project.
- DxDesigner File>Export provides a huge range export options:
    - Netlist (NL) formats to drive third party layout systems such as KeyIn NL, Cadence Allegro NL, RINF NL.
    - NLs to drive analysis tools such as VHDL, Verilog, and Analog.
    - Import to manufacturing viewers and readers such as EDIF schematic & EDIF Netlist.
    - to export front end schematic project (Remote Development) for remote working away from project server, terminator, test point, connector) would be assigned to each pin to allow proper ordering.

Section

10

Migration Process & Training

10.1 Will we need different hardware or network performance to use DxDesigner?

In the basic single-user mode of operation, DxDesigner does not require any infrastructure beyond what is used for DC.

To support multi-user collaboration a collaboration server will be required with good network speed between server and clients. It is also recommended that the design project data and central library be located on this server.

An overview presentation of Expedition Enterprise infrastructure and collaboration server is available on SupportNet at the link below.
http://supportnet.mentor.com/reference/tutorials/index.cfm?id=10755

10.2 Will training be required when moving from Design Capture to DxDesigner?

Although DxDesigner is intuitive and easy to use, training is recommended as DxDesigner is similar in many areas but remains a different product and in some cases different methods are employed to achieve the desired result.

TRAINING OPTIONS
Training options include the following:

- A three-day formal training class is available at a Mentor Graphics facility or online. For more information please contact your local Mentor sales office or email to education_services@mentor.com
- A self-paced DxDesigner tutorial is available on SupportNet at the link below
  http://supportnet.mentor.com/reference/appnotes/index.cfm?id=MG579468

- Specifically for smaller teams the Mentor training department is providing free migration-focused DxDesigner training for Design Capture customers at no cost. More information can be found here: http://www.mentor.com/training_and_services/training/schedule

- Mentor is providing a Virtual Lab, which allows users to practice migrating a library and design in a cloud-based test environment following a documented process. This can be found here : http://www.mentor.com/go/DC2DX

10.3 Can DC/Expedition projects be moved to DxDesigner/Expedition?

Yes. A no-cost project translator is included in the standard Expedition Enterprise installation media. The project translator has been available for many years and has been used by many customers who have moved to DxDesigner.

TRANSLATION PROCESS DOCUMENTS

The translation process is fully documented on SupportNet. Key documents are listed below.

- Migration Planning Guide
  http://iweb.mentor.com/migration/7.9.4/index.html

- Design Capture-Expedition to DxDesigner-Expedition Translation Checklist
  http://supportnet.mentor.com/docs/201206008/docs/pdfdocs/dc2dx_trans_checklist.pdf

- Design Capture/Expedition PCB to DxDesigner/Expedition PCB Translation Guide
  http://supportnet.mentor.com/docs/201206008/docs/pdfdocs/dc2dx_trans_process.pdf

- Design Capture to DxDesigner translation process tutorial
  http://supportnet.mentor.com/reference/tutorials/index.cfm?id=10574

- Design Capture to DxDesigner Concepts and Terms Cross Reference
  http://supportnet.mentor.com/reference/appnotes/index.cfm?id=10854

10.4 My company has various schematic capture products. Can these schematics be imported into DxDesigner?

- Many schematic formats can be easily imported in to DxDesigner via the Import process. These DxDesigner import tools are provided as part of the standard installation at no additional cost.
- The 3rd party schematic import support includes Cadence OrCAD, Cadence ConceptHDL, Altium, CadStar and others. Please check SupportNet for versions supported, which can change with each release.

10.5 Today DC is used to interface with a 3rd party layout tool via the DC Connections option. Can this type of schematic be moved to DxDesigner?

- The schematic can be translated to the DxDesigner “Integrated flow” to be used with Expedition layout wherein a 3rd party generic net list maybe created.
- The full DxDesigner license “DxDesigner for Expedition” does have the option to create a DxDesigner “Independent NetList project” interfacing with 3rd party layout systems however this option would require further consideration of symbol library differences.
    - A DxDesigner “Independent NetList project” can be imported to DxDesigner “Integrated flow”
- The DC schematic cannot be translated to a DxDesigner “Independent NetList project”, as the translator automatically creates an integrated-flow project.

Index

Index

A

Automation and Scripting Community, 22

C

CES, 9, 10
Concepts and Terms, 26
cross probing
  licenses, 7

D

DC Connections, 27
DC2DX Migration Community, 8
Design Capture
  history and future plans, 6
Design Outsourcing, 24
DxDesigner
  history and future plans, 6, 7
  platform support, 7
DxDesigner Automation, 22
DxPDF, 23

F

File>Export, 24
formal training, 25

I

Independent NetList Projects, 27

M

Migration Planning Guide, 8, 26
migration-focused training, 26

N

Net Class/Net Properties, 9

P

PDF generation, 23

R

Report Writer Community, 8

S

schematic import, 27
self-paced tutorial, 25

T

training options, 25
Translation Checklist, 26
Translation Guide, 26
TRANSLATION PROCESS DOCUMENTS, 26
Translation Process Tutorial, 26

U

Undock/Dock, 24

V

virtual lab, 26